

# ANSYS Taylor-Couette Flow between Rotating Cylinders - Physics Setup

Author(s): Sebastian Vecchi, ANSYS Inc

## Problem Specification

1. Pre-Analysis & Start-Up
2. Geometry
3. Mesh
4. Physics Setup
5. Solution/Results
6. Verification & Validation

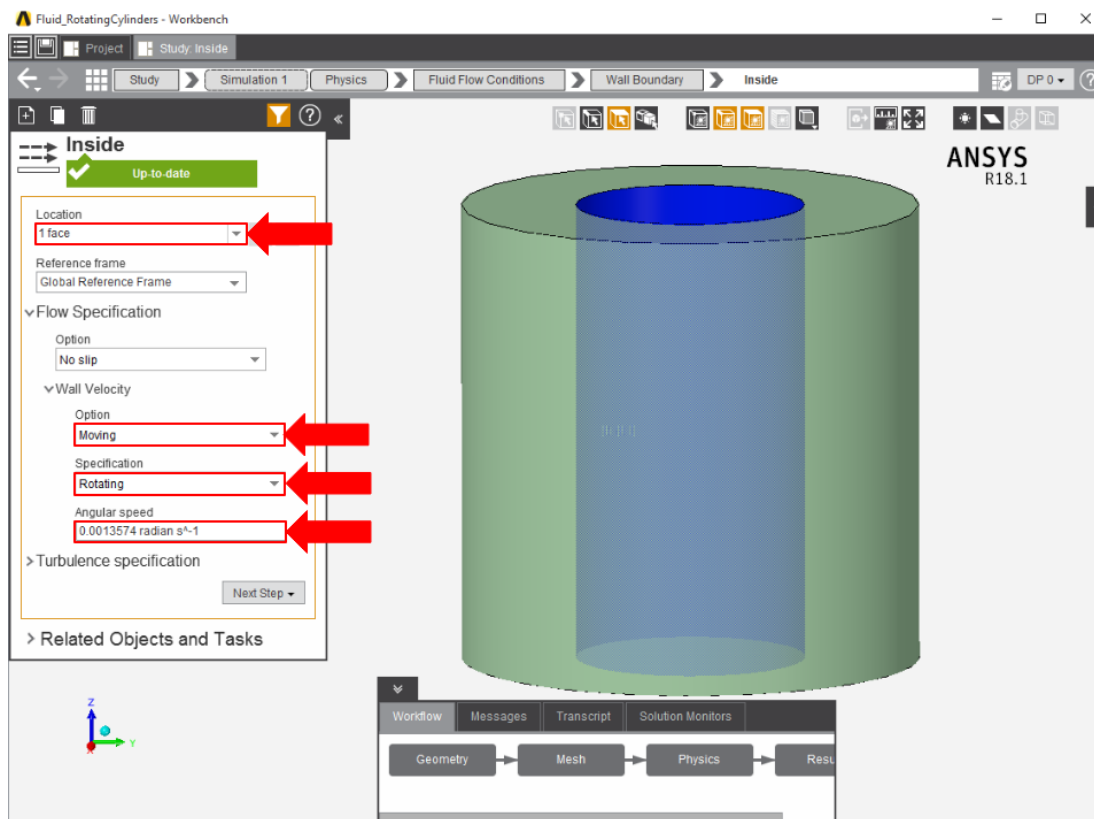
## Physics Setup

### Change Materials

In order to make the simulation more realistic, the material needs to be changed from Air to Water. Select **Material Assignments** then change the **Material** to **Water (Material Samples)**.

### Boundary Conditions / Forces

In this problem, we are asked to move the two cylinders with different speeds in the same direction. This can be done by assigning wall conditions to both faces of the cylinders in contact with the flow volume. First, using the **Face selection** tool, choose the **inside face** of the flow volume. Then, select the **Add** drop down menu next to **Fluid Flow Conditions** and select **Wall**. A panel will appear with the cylinder face as the location of the wall. From here, change the **Wall velocity** from **Stationary** to **Moving** and **Specification** to **Rotating**. The **Angular speed** of the inner wall of the flow volume is defined in the problem specification to be  $0.00135736 \text{ [radian s}^{-1}\text{]}$ .



Now that the wall boundary has been created for the inner flow volume face, making the wall boundary for the outer flow volume face will be similar. Select the outer face of the enclosure and select **Wall** from the **Add** drop down menu.

Now that both rotating cylinders have been defined, a free slip condition will be put on both ends of the flow volume. This is also done using the **Wall** tool. Pick **Wall** from the **Add** drop down menu. Select the top and bottom faces of the flow volume with the **Face selection** tool. Add these faces to the **Location** and then change **Flow specification** to **Free slip**. Use the picture below for guidance.

**Go to Step 5: Solution/Results**

[Go to all ANSYS AIM Learning Modules](#)